PSZ 19:16 (Pind. 1/97)

UNIVERSITI TEKNOLOGI MALAYSIA

BORANG PENGESAHAN STATUS TESIS

JUDUL: **FINITE ELEMENT ANALYSIS ON FLUSH END PLATE CONNECTION USING LUSAS SOFTWARE**

SESI PENGAJIAN: 2005/2006

Saya **CHE SOM MOHD YUSOFF**
(HURUF BESAR)

mengaku membenarkan tesis (~~PSM/~~Sarjana/~~Doktor Falsafah~~)* ini disimpan di Perpustakaan Universiti Teknologi Malaysia dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Teknologi Malaysia.
2. Perpustakaan Universiti Teknologi Malaysia dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

☐	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
√	TIDAK TERHAD	

Disahkan oleh

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:
NO.45 JALAN BUKIT MEWAH 48, TAMAN BUKIT MEWAH, 4300 KAJANG SELANGOR.

PROF. MADYA DR.SARIFFUDDIN BIN SAAD
Nama Penyelia

Tarikh: **15 MEI 2006** Tarikh: **15 MEI 2006**

CATATAN:
* Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.
♦ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

"I declare that I have read through this project report and to my opinion this project report is adequate in term of scope and quality for the purpose of awarding the degree of Master of Engineering (Civil – Structure)".

Signature : ....

Name of Supervisor : ASSOC. PROF. DR. SARIFFUDDIN BIN SAAD

Date : 1□MEI 2006

FINITE ELEMENT ANALYSIS ON
FLUSH END PLATE CONNECTION USING
LUSAS SOFTWARE

CHE SOM MOHD YUSOFF

A project report submitted in partial fulfillment of the requirement for the award of the degree of Master of Engineering (Civil – Structure)

Falculty of Civil Engineering
Universiti Teknologi Malaysia

MEI 2006

"I declare that this project report entitled " Finite Element Analysis on Flush End Connection using LUSAS software" is the result of my own research except as cited in the references. The report has not been accepted for any degree and is not concurrently submitted in candidature of any other degree"

Singnature	:	....
Name	:	CHE SOM MOHD YUSOFF
Date	:	1□MEI 2006

DEDICATION

To husband and children

Thank you for your support

&

To friends

thank you for eveything

ACKNOWLEDGEMENT

Alhmadulilah, Praise to Almighty Allah for the blessing and His permission, I am able to complete my master project.

I wish to extent my greatest thank you and gratefulness to my supervisor, Assoc. Prof. Dr. Sariffuddin Bin Saad for his valuable guidance, advice and suggestions throughout this project. With his effort and concern, I am able to complete my project. Thank you also to Ir. Che Husni Hj Ahmad from Perunding Arne, for his help and advise.

I am also grateful to my most beloved family for their love and kindness towards me and for their strong support during my study period. I also wish to thank my parents for their never ending support for my family and me.

Finally, a lot of thank you to all lecturers and staff of Faculty of Civil Engineering, University Teknologi Malaysia, Skudai, Johor and also for all my friends, student of postgraduate of Structural and Material Department for their support and cooperation throughtout my study.

Thank you very much.

ABSTRACT

This paper present and discuss the finite element method as an alternative method to investigate the behaviour of the flush end plate. The finite element results are compared with the results of the experimental tests taken from literature. The three dimensional materially static non-linear finite element analys is using **LUSAS** modeller was perform the bolted flush end plate connection. The moment-rotation curve was plotted from the result and the curve was superimpose with the experimental curve to check the accuracy of the analyses method. It was found that the two curves coin-side quite well with each other. The difference in values of the moment resistance of the two curves is 15..62 %. In this research also found that a 3.125% increase in the end plate thickness produces a 3.03% the moment of resistance of the connection. The results of this study shows an agreement with previous researchers that the connecti behaves in between pinned and fully rigid and posses some rotational stiffness.

ABSTRAK

Tesis ini membincangkan mengenai finite element method sebagai satu alternatif untuk mengkaji kelakuan sambungan plat hujung sedatar. Keputusan dari finite element method dibandingkan dengan keputusan ujian yang telah dijalankan sebelum ini. Model yang dibuat dalam perisian komputer adalah dalam bentuk tiga dimensi dengan menggunakan LUSAS modeler. Graf momen–putaran dilakarkan dan dibandingkan dengan graf momen-putaran yang sedia ada dari ujian makmal. Dari analisa didapati bahawa lengkungan antara ujian makmal dan computer adalah bersentuhan antara satu sama lain. Perbezaan bagi momen rintangan adalah 15.62 %. Untuk plat yang berlainan ketebalan juga dapat dianalisa dan keputusannya adalah 3.125 % dan momen rintangan juga meninkat sebanyak 3.03 %. Keputusan dari kajian ini menunjukan persamaan dengan penyelidik-penyelidik terdahulu yang mana perlakuan sambungan adalah di antara sambungan pin dan tegar serta sedikit kekukuhan putaran.

CONTENTS

CHAPTER			PAGE
	TITLE		I
	DECLARATION		ii
	DEDICATION		iii
	ACKNOWLEDGEMENT		iv
	ABSTRACT		v
	ABSTRAK		vi
	CONTENTS		vii
	LIST OF TABLE		xi
	LIST OF FIGURE		xii
1	**INTRODUCTION**		1
	1.1	Problem Statement	1
	1.2	Problem Study	1
	1.3	Objective of study	2
	1.4	Scope of study	2
	1.5	Significance of Study	3

2	**LITERATURE REVIEW**		
2.1	Introduction		4
2.2	Behaviour of semi-rigid connection		5
2.3	Types of semi rigid		
	2.3.1	Single Web-Angle Connection and Single Plate Connection	7
	2.3.2	Double Web-Angle Connection	8
	2.3.3	Top and seat-angle connection with double web angle	9
	2.3.4	Top and seat-angle connection without double Web angle.	9
	2.3.5	Extended end plate and flush end plate connection	10
	2.3.6	Header-plate connection	12
2.4	Moment Rotation (M-Φ) Characteristic		13
2.5	Connection Classification		13
2.6	Analysis of connection		17
	2.6.1	Experiment set-up	17
	2.6.2	Finite Element Method	18
	2.6.3	Advantages and Disadvantages	

		of the FEM	19
	2.6.4	Background of LUSAS	19
	2.6.5	LUSAS Software Characteristic	20
	2.6.6	Procedure Analysis According to LUSAS Software	20
2.7	Research Paper Studies		21
	2.7.1	Finite Element Analysis of structural Steelwork Beam to Column Bolted Connection.	22
	2.7.2	Parametric analysis of steel bloted end plate connections using finite element,	23
	2.7.3	Development of moment-rotation model equations for flush end-plate connection,	23
	2.7.4	Experiment assessment of the ductility of extended end plate connection	24
	2.7.5	Non-linear Analysis of symmetric Flush end plate Bolted beam to column Steel Connection.	25
	2.7.6	Prestasi sambungan pada paksi major menggunakan keratan keluli tempatan.	25

3	**METHODOLOGY**		26
3.1	Introduction.		26
3.2	Finite Element Model		27
3.3	Constitutive Models		28
	3.3.1	Isotropic Model	28
	3.3.2	Stress Potential Model	28
3.4	Type of Element		29
3.5	Load and boundary Condition		31
3.6	Non-Linear Analysis		31
	3.6.1	Materially Non-linear Analysis	31
	3.6.2	Geometrically Non-linear Analysis	32
	3.6.3	Non-linear FEA Procedures	33
	3.6.4	Increment Procedure.	35
	3.6.5	Incremental Loading	36
	3.6.6	Solution termination	36
	3.6.7.	Non linear Convergence Criteria	37
4	**RESULTS AND DISCUSSIONS**		

	4.1.	Introduction	38
	4.2	Non-linear Analysis Result.	38
		4.2.1 Dimensions and geometric Parameter	38
5	**CONCLUSION AND RECOMMENDATION FOR FUTURE RESEARCH**		
	5.1	Conclusion	50
	5.2	Recommendation for future reference.	
6	**REFERENCE**		52

LISTS OF TABLE

TABLE NO.	TITLE	PAGE
3.1	The element types used to the model	30
4.1	Dimension (in mm) of the flush end plate connection	40
4.2	Tabulation of displacement for point A and B	43
4.3	Tabulation of moment and Rotation(ϕ_j)	44

LISTS OF FIGURES

FIGURE NO.	TITLE	PAGE
2.1	Rotational deformation of a connection	6
2.2	Moment – rotation for connection	7
2.3a	Single Web Connection	8
2.3b	Single Plate	8
2.4	Double Web Angle	8
2.5	Top and seat angle connection with double web	9
2.6	Top and seat-angle connection without double Web angle	10
2.7a	Extended end plate for tension side	11
2.7b	Extended end plate both side (compression/ tension)	11
2.7c	Flush end plate	12
2.8	Header-plate connection	12
2.9	Classification of connection by strength	14
2.10	Classification of connection by rigidity	15

2.11	Classification of connection by ductility.	16
2.12	Typical experimental set-up	17
2.13	A finite element model representing a real engineering problem	18
3.1	Typical Idealized Uni-axial stress-stain Relationship for steel.	32
3.2	Geometric Non-Linearities	33
3.3	Increment iterative method	35
4.1	Model Geometric	39
4.2	Defected shape Of the overall model	41
4.3	Point A and B for displacement	42
4.4	Graph M- ϕj	44
4.5	Graph M- ϕ_j	46
4.6	Graph M- ϕ_j	47
4.7	Graph M- ϕ_j	48

LIST OF SYMBOLS

M	Moment
ϕ	Rotation
ε	Strain
σ	Stress
dy	Displacement in Y
dz	Displacement in Z
k	Spring stiffness
v	poisson ratio
E	Modulus Young

CHAPTER I

INTRODUCTION

1.1 Problem Statement

Connections in a steel frame structure are very important elements in transferring load such as gravity, wind and seismic load. Therefore it is important to obtain an understanding at their structural behaviour. In the past, experimental approach has been used widely to study the behaviour of connections. But these are expensive and sometimes difficult to perform. An alternative is to use finite element analysis but the results need to be calibrated against good experimental test data. Therefore in this project it was proposed to carry a finite element investigation to study the behaviour of a flush end plate connection.

1.2 Problem of Study

The overall behaviour of connection is importance in analysis and design of structures. In design, a connection is considered as pin or rigid where a pin connection only take the axial force while the rigid connection resists the moment without any rotation.

In real a situation, a connection behaves in between the two cases above which is call semi-rigid. The most accurate method to study the non-linear behaviour of a connection is to fabricate the full scale connection and test these to fail. Unfortunately this is time consuming, expensive to undertake and has the disadvantage of only recording strain readings at pre-defined gauge locations on the test connection.

A three dimensional materially static non-linear finite element analysis approach has therefore been developed as an alternative method of connection appraisal.

1.3 Objectives of Study

Main objective of this research is to use the finite element analysis using LUSAS software [1] to model and analyse a flush end plate connection. The moment-rotation curve will be plotted and compared the result of an experimental test.

1.4 Scope of Study

The experimental work reported by Husin [2] focused mainly on flush end plate bolted connection. The plate has 4 holes and M20 bolt were used. The end plate thickness was 12 mm.

The column size was 200 x 200 x 56.2 – 3 m and the size of the beam was 250 x 125 x 25.1 – 1.5m. Static loads were applied at 1.3 m from the face of the column.

LUSAS software will be used to model the connections. The result from the finite element analysis, mainly moment-rotation curve, will be compared with the existing experimental result.

1.5 Significance of Research

The research significance to be obtained from this study will be the results and analysis of the behaviour of beam to column flush end plate connection. It is necessary to compare the moment –rotation curve of the result from the finite element analysis and that of the experimental testing. The aim is to determine the accuracy of the analytical method and to verify the predicted strength of the flush end plate connection.

CHAPTER II

REVIEW OF LITERATURE

2.1. Introduction

Conventional analysis and the design of steel frameworks are performed using the assumption that the connections are either fully rigid or ideally pinned. The assumption of the fully rigid connection implies that no relative rotation of the connection occurs and the end moment of the beam is completely transferred to the column. On the other hand, the pinned connection implies that no restraint for rotation of the connection exists and the connection moment always zero. However, as is evident from experimental observations, all beam to column connection used in the current practice posses some stiffness that falls between the two extreme cases of fully rigid and ideally pinned [3].

The connection that behaves in between the pinned and rigid is called the semi rigid connection or partially restrained (type PR as specified by the AISD LRFD-1993). Therefore, to establish the guideline for design of this type of connection, its is necessary to know the behaviour of the actual beam to column connections and to formulate an appropriate connection model for use in the design and analysis of semi-rigid frames.

Despite numerous years of extensive research, particular in the 197□'s, no fully agreed design method exists. Many areas of connection behaviour still require investigation. More recently Bose, Sarkar and Bahrami used [4] FEA to produce moment rotation curves. Then Bose, Youngson and Wang [5] reported on 18 full scale tests to compare moment resistance, rotational stiffness and capacity. The latest design method utilizes plastic bolt force distribution to create an increase moment connection capacity and reduced column stiffening. In 1995 when the SCI and the BCSA produced the Green Book guide, based on the EC3 design model, the editorial committee felt a number of areas, particularly bolt force distribution and compression flange overstress required further investigation.

2.2 Behavoiur of semi rigid connection

A connection is a medium through which forces and moments are transferred from beam to column. The connection subject to axial forces, shear force, bending moment and torsion. The effect of torsion can be excluded, shear and axial forces also very small compared to bending moment. Consequently, for practical purposes, only the effect of moment on the rotational deformation of connection needs to be considered. As depicted in Fig 2.1 below the connections rotates by the amount ϕ_r when a moment M is applied. The angle ϕ_r corresponds to the relative rotation between the beam and the column at the connection. Therefore, the in plane behaviour of semi-rigid connection is represented by the M-ϕ_r rotation, referred to as moment-rotation behaviour.

The moment-rotation behaviour of the variety of commonly used semi-rigid connection is shown in Fig. 2.2. All types of connection exhibit non-linear moment-rotation behaviour that fall between the two extreme cases of pinned jointed and rigid connection. The moment-rotation curves of all types of connection are non-linear over the entire range of loading. The nonlinearity of connection behaviour due to a number of factors such as material discontinuity of connection subssemblage, local yielding of some component part, local buckling of the plate element and so on. Semi-rigid connections are generally very ductile connection which offer a significant contribution towards the ultimate capacity of the structure.

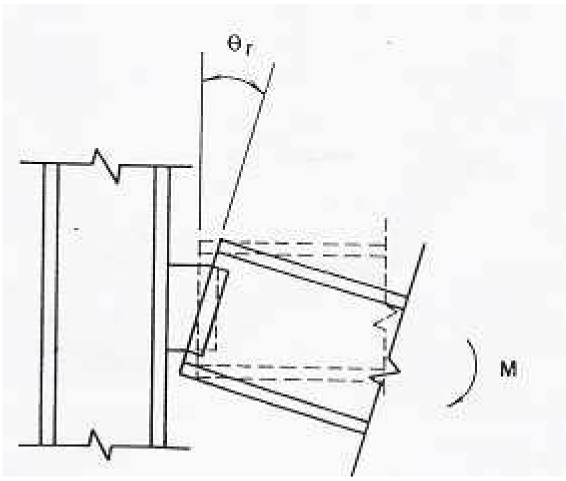


Figure 2.1 Rotational deformation of a connection

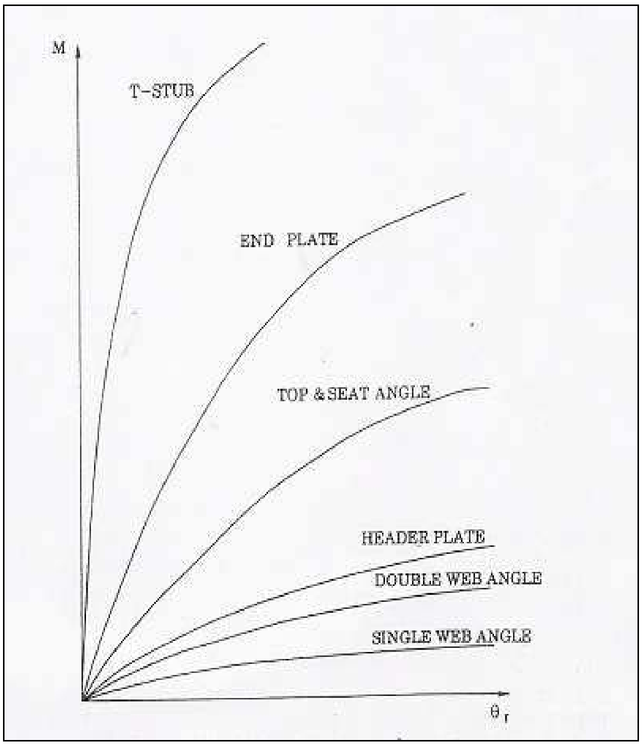


Figure 2.2 Moment – rotation for connection

2.3 Types of semi rigid

2.3.1 Single Web-Angle Connection and Single Plate Connection

A single web-angle connections consists of an angle either bolted or welded to both the column and the beam web, as shown Fig. 2.3a. On the other hand, a single plate connection uses the plate instead of the angle. This connection type requires less material than a single web angle connection (Fig. 2.3b). Generally, the single web-angle connection has moment rigidity equal to about one half of the double web-angle connection. The single plate connection has rigidity equal to or greater than the single web-angle connection because one of the plate in the single plate connection is fully welded to the column flange.

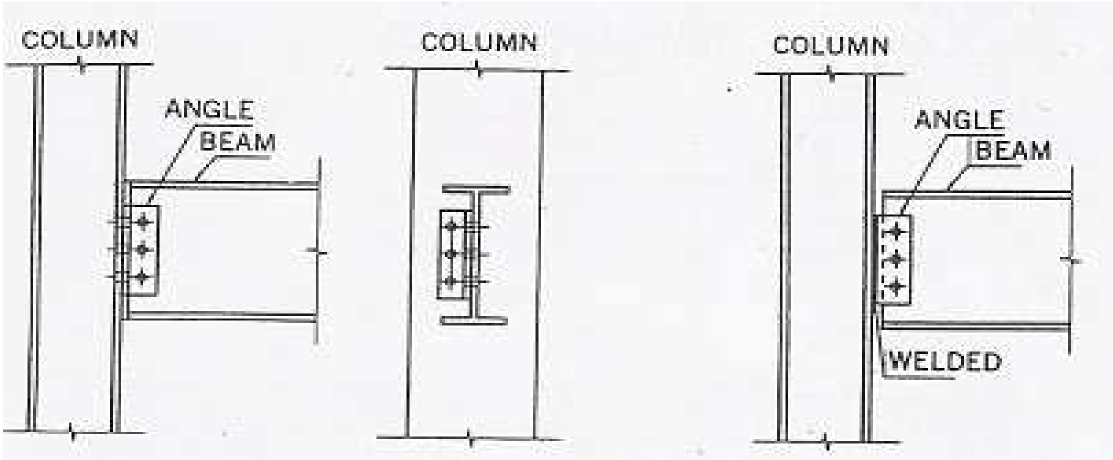


Figure 2.3a Single Web Connection Figure 2.3b Single Plate

2.3.2 Double Web-Angle Connection

A double web-angle connection consists of two angles, bolted to column and beam web, as shown in Fig 2.4. Although the connection rigidity of this type is stiffer than those of single web-angle and single plate connection, the AISC consider this type as type 2 connection (simple or shear connection).

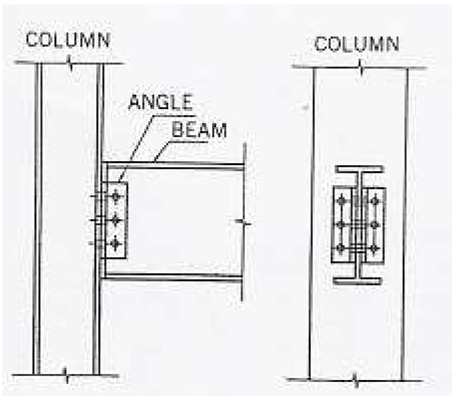


Figure 2.4 Double Web Angle

2.3.3 Top and seat-angle connection with double web angle

This type of connection is a combination of top and seat-angle connection and double web-angle connection. A typical top and seat angle connection with double web angle is illustrated in Fig 2.5. Double web angles are used to improve the connection restrain characteristic of top and seat angle connections and for shear.

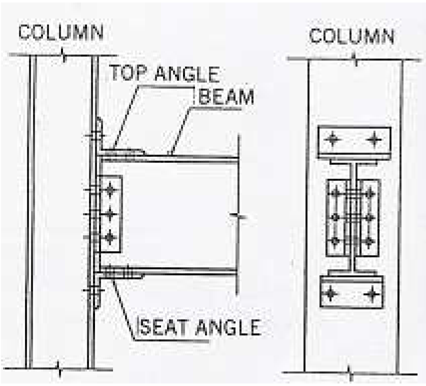


Figure 2.5 Top and seat angle connection with double web

2.3.4 Top and seat-angle connection without double Web angle.

A typical top and seat angle connection is shown below (Fig 2.6). The AISC described this type of connection as follows:

(a) the top angle is used to provide lateral support of the compression flange of the beam.

(b) The seat angle is to transfer only the vertical reaction of the beam to the column and should not give a significant restraining moment on the end of the beam. However according to the experimental results, this connection can transfer not only the vertical reaction, but also some end moments of the beam to the column.

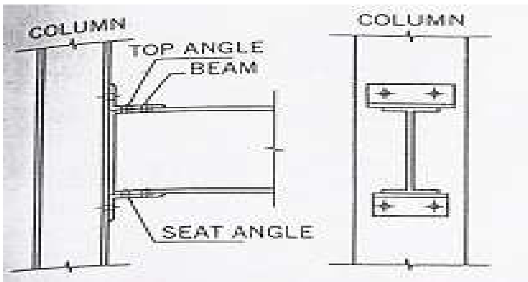


Figure 2.6 Top and seat-angle connection without double Web angle

2.3.5 Extended end plate and flush end plate connection

In general, an end plate is welded to the beam along both the flanges and web in the fabricator's shop and bolted to the column in the field. The end plate connection has been used extensively since 196□s. The extended end plate connection are classified into two types;

a) extended on the tension side (Fig.2.7a) or

b) extended on the both side (compression and tension)(Fig. 2.7b)

A typical flush end plate shown in the Fig.2.7c. The extended end plate connection on both sides (see Fig. 2.7b) is preferred when the connection is subject to moment due to severe earthquake loading.

Although the flush end plate is weaker than extended end plate, this connection is often used for roof detail. The behaviour of an end plate connection depend on whether the column flange near the connection is stiffened or not. The stiffeners of the column flanges act to prevent flexural deformation of the column flange, thereby influence the behaviour of the plate and fastener.

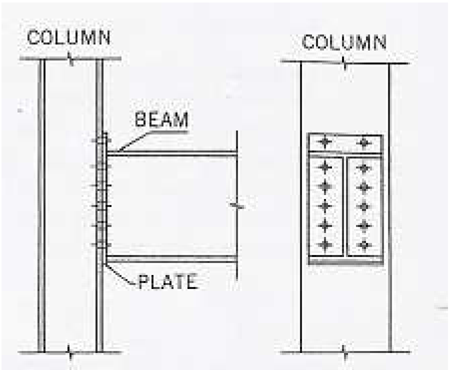


Figure. 2.7a - Extended end plate for tension side

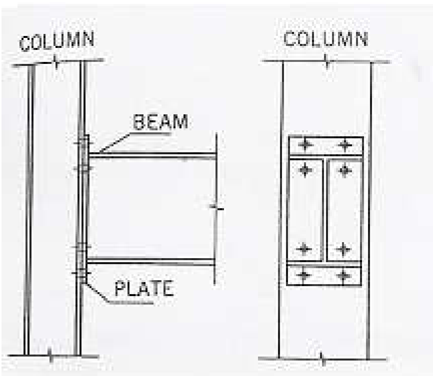


Figure. 2.7b Extended end plate both side (compression/ tension)

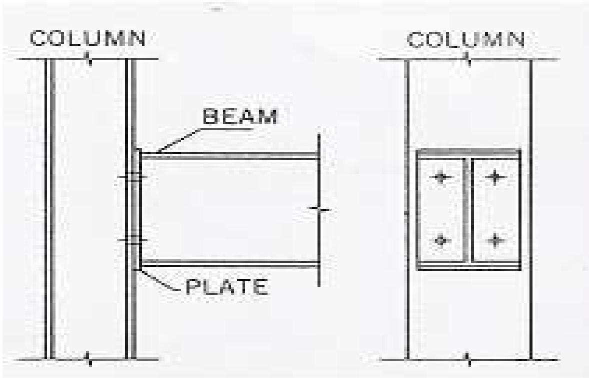


Figure 2.7c Flush end plate

2.3.6 Header-plate connection

A header plate connection consists of an plate, whose length is less than the depth of the beam, welded to the beam web and bolted to the column, as shown in Fig. 2.8. The moment rotation characteristics of this connection are similar to those of the double web-angle connection.

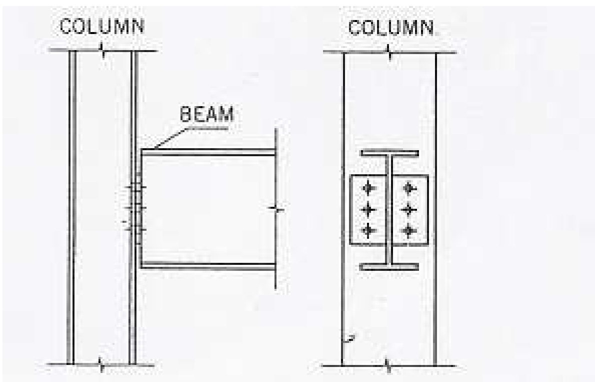


Figure 2.8 Header-plate connection

2.4 **Moment Rotation (M-ϕ) Characteristic**

The main structural elements of steel framed multi-storey structures are the columns, the beams and their connections. Conventionally the beam-to-column connections are considered to be either pinned or rigid. In the case of pinned or 'simple' connections, the frames have to be stabilised by appropriate bracing systems. Such frames are named braced frames by Eurocode 3.

The term 'rigid' in this context implies that the connection is capable of resisting moments with a high stiffness, i.e., the connection flexibility has a negligible influence on the distribution of movements in the frame connections. When the connections are rigid, the overall stability may be provided by the frame itself without the inclusion of specific bracing systems.

2.5 Connection Classification

A further classification of connection is related to their strength. A 'full-strength' connection is a connection that can at least develop the bending strength of the elements it connects. A 'partial-strength' connection has a lower design strength than that of the elements it connects. Fig. 2.9 shows connections 1, 2 and 4 are full strength connection where the moment resistance is at least equal to that of the member. Connections 3 and 5 are partial strength connections as the moment resistance is less than that of the member and connection 6 is pinned joint.

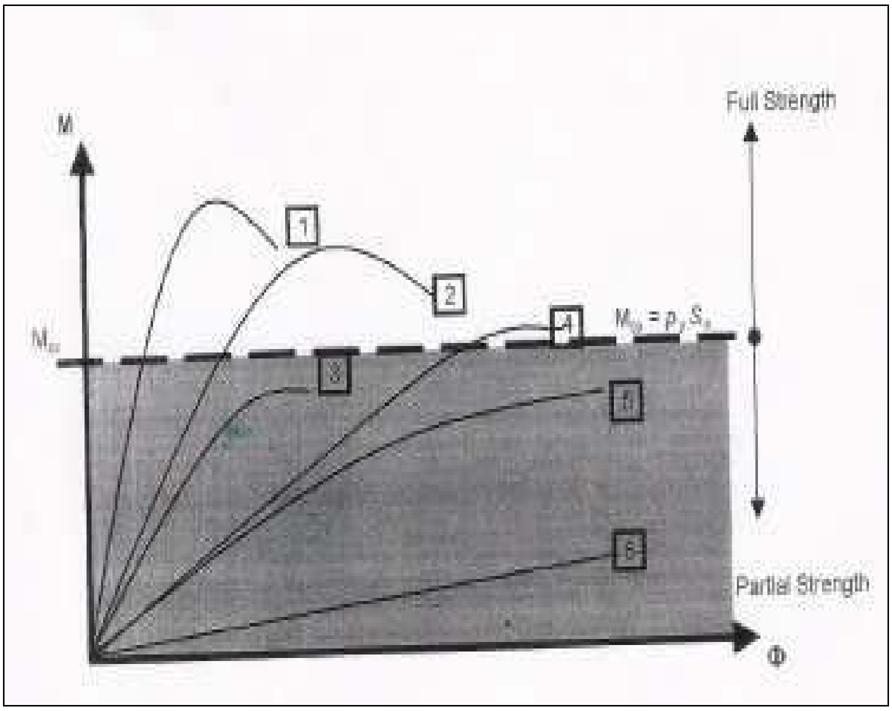


Figure 2.9 : Classification of connection by strength

Connections 1,2,3 and 4 shown in Fig 2.1 below are rigid connections which are stiff enough for the effect of its flexibility on the frame bending moment to be neglected. Connection 5 is a semi rigid connection. BS 595 indicates that the slope of dividing line between semi-rigid should be taken as 2EI/L.

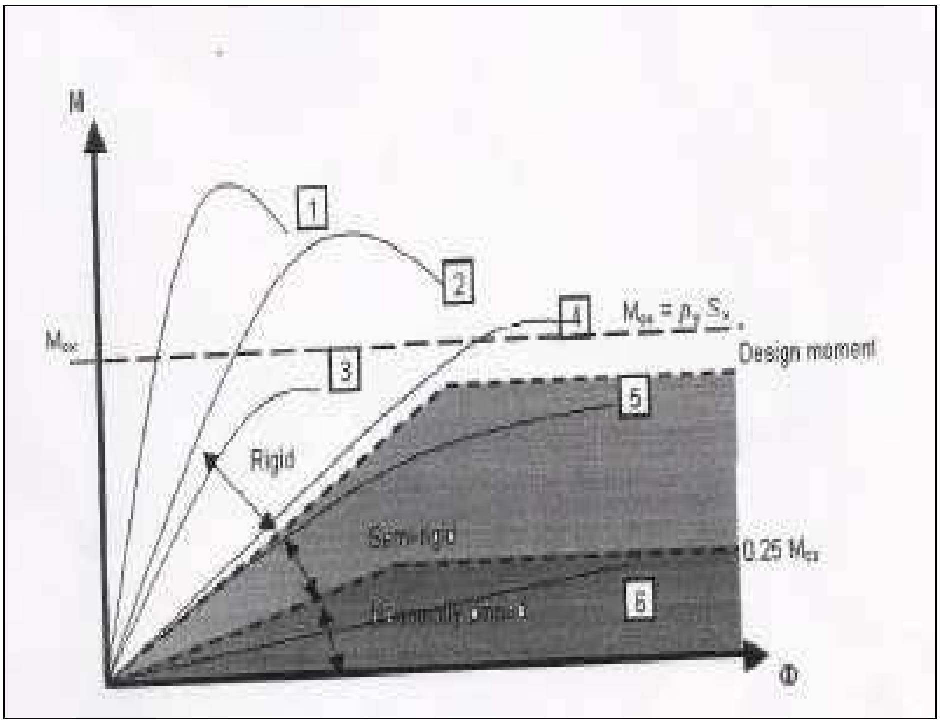


Figure 2.1□: Classification of connection by rigidity

The rotation capacity of a moment-resisting connection can also be important. For example a beam with partial-strength end connections can be designed plastically if the connection rotation capacity is sufficient to ensure the development of an effective hinge are considered as ductile connection as shown in Fig 2.11. The connections 2, 4 and 5 are ductile while connection 1 is not ductile. Connection 6 is a pinned connection.

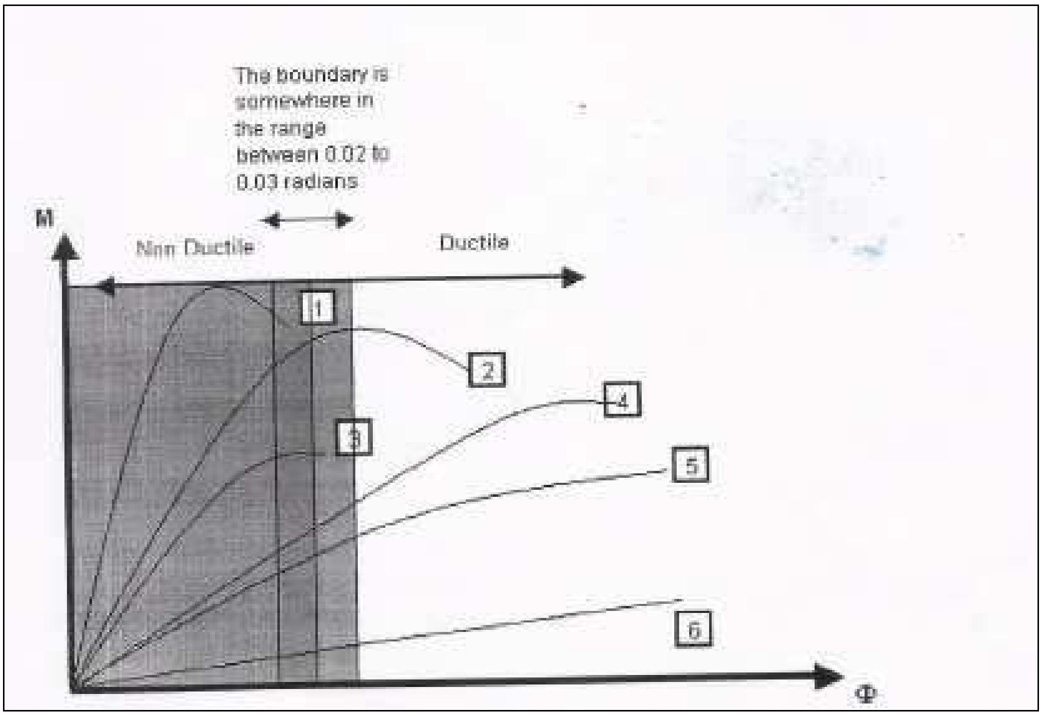


Figure 2.11: .Classification of connection by ductility.

2.6 ANALYSIS OF CONNECTIONS

2.6.1 EXPERIMENTAL SET-UP

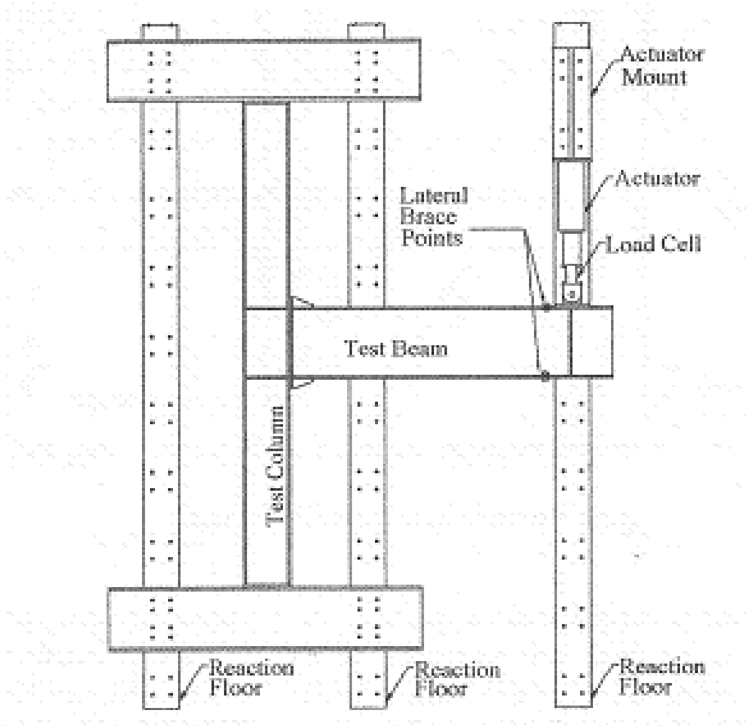


Figure 2.12 Typical experimental set-up[2]

Fig. 2.12 shows the detail arrangement of the flush end plate connection involving a column and flat web beam. This connection was tested at University Technology Malaysia Skudai in Johor [2]. The M-ϕ curve for this connection is available for comparison with the M-ϕ result of the present work.

2.6.2 Finite Element Method

Basic ideas of the finite element method originated from advances in aircraft structural analysis. In 1941, Hrenikoff presented a solution of elasticity problems using the "frame work method." Courant's paper, which used piecewise polynomial interpolation over triangular sub regions to model torsion problems, appeared in 1943 [1]. Turner, et al. derived stiffness matrices for truss, beam and other elements and presented their findings in 1956 [1]. The term finite element was first coined and used by Clough in 196□[1].

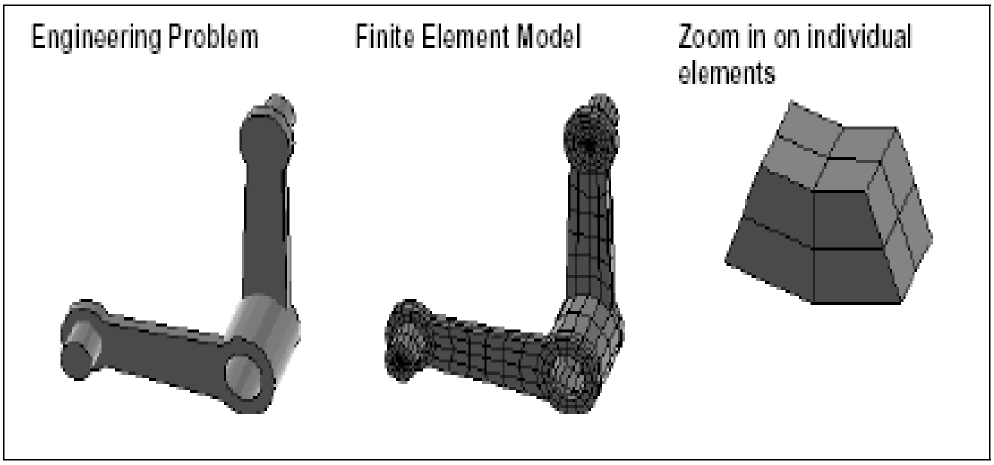


Figure 2.13: A finite element model representing a real engineering problem

Fig. 2.13 shows a finite element model of a real engineering problem.

Solution of finite element can be summarised as involving the combination of several nodes to produce a small total unit that balance each other. The combination of many such units then form a model.

2.6.3 Advantages and Disadvantages of the FEM

The finite element method has several advantages;

a) it can model complicated geometry.
b) it can be programmed into a computer software and this may avoid the need to perform laboratory test.
c) it can analyse difference element characteristics in a matrix system.
d) it can handle non-linear behaviour involving high deformation and non-linear material.
e) it can take the boundary conditions into account.

But the finite element method has several disadvantages, i.e:

a) it needs a professional to model an exact situation.

b) it needs a computer with a high processor to run the analysis smoothly.

c) the method is mesh sensitive so that it is necessary to try various element densities at critical areas before the results can be accepted.

2.6.4. Background of LUSAS

LUSAS is computer software for structural analysis based on the finite element method; it was built by FEA Limited at United Kingdom. This software can be combined with the CAD function to show the modelling shapes, stress distribution sand post-processing shape changes.

LUSAS is an associative feature-based modeller. The model geometry is entered in terms of features which are sub-divided (discretised) into finite elements in order to perform the analysis. Increasing the discretisation of the features will usually result in an increase in accuracy of the solution, but with a corresponding increase in solution time and disk space required. The features in LUSAS form a hierarchy that is volumes are comprised of surfaces, which in turn are made up of lines or combined lines, which are defined by points.

2.6.5 LUSAS Software Characteristic

LUSAS software can analysis and organise complex structure problems and shapes including 3 dimensional structures. This software also can be used in dynamic structural analyses with temperature changes. LUSAS software can solve problems up to 5□□number of elements.

2.6.6 Procedure Analysis According to LUSAS Software

There are 3 steps in the finite element analysis using the LUSAS software, which are as follows:

a) Pre-processing phase

Pre-processing involves creating a geometric representation of the structure, then assigning properties, then outputting the information as a formatted data file (.dat) suitable for processing by LUSAS.

b) Finite Element Solver

Sets of linear or nonlinear algebra equations are solved simultaneously to obtain nodal results, such as displacement values at different nodes or temperature values at different nodes in heat transfer problems.

c) Result-Processing

In this process, the results can be processed to show the contour of displacements, stresses, strains, reactions and other important information. Graphs as well as the deformed shapes of a model can be plotted.

2.7 Research Paper Studies

The research papers by a number of authors who investigated on moment-rotation behaviour of end plate connections are presented here. Most of them deal with experimental testing and followed by finite element validation using software application.

2.7.1 Finite Element Analysis of structural Steelwork Beam to Column Bolted Connection.[6]

The research conducted 5 series of full scale tests were conducted using self straining frame in the heavy structure laboratory at the University of Teeside. Beside the full scale testing, the finite element model were developed using MYSTRO and LUSAS to moment resistance of extended end plate connection where the result was compared with that of the Green Book Theoretical Capacity.

Jim Butterworth concluded that finite element and laborotary results were consistent. He also indicated that the Green Book design theory underestimated the bolts forces in the top rows of the connection and overestimated the forces in the lower row. It was reported that the finite element analysis of extended end-plate connection can be seen to provide advantages in term of time and expense over full scale testing and can produce a more complete picture of stress, strain and forces distribution.

2.7.2 Parametric analysis of steel bloted end plate connections using finite element.[7]

This paper presented and discussed results of analyses on the behaviour of bolted extended end plate connection using finite element modeling tools. These results of the analyses were calibrated to that of the experimental results. The analytical models took into account material and geometricl non-linearities as well large displacements of the end plate, and forces on the bolts showed satisfactory agreement.

2.7.3 Development of moment-rotation model equations for flush end-plate connection.[8]

The paper reported the development of moment rotation model equations for flush end-plate connection. The author presented the development of Ramberg-Osgood and three Parameter Power Model predicted equation for the moment rotation (M - ϕ) behaviour of flush end-plate. They indicated that the M - ϕ characteristic are related to stiffness, strength, and the geometric variables i.e plate thickness, bolt diameter, bolt pitch etc. A three dimensional non-linear finite element model was generated using the finite element analysis software package, ANSYS.

The accuracy of the finite element result was verified by using experimentally test that available. The ANSYS software was used to analyse the 34 test cases for (M - ϕ) data, which were curve fitted Ramberg-Osgood and Three Parameter Power model equations to obtain defining parameters. Regression equations were develop for the prediction models for flush end-plate connection. It was shown that both model predicted the (M - ϕ) closely, with more accurate model being the Three parameter Power Model.

2.7.4 Experiment assessment of the ductility of extendedplate connection.[9]

The paper present the experimental result of 8 statically loaded extended end plate moment connection. The parameter investigate were endplate thickness and steel grade. The result shows that an increase in end plate thickness results in an increase in the connection flexural strength and stiffness and a decrease in rotational capacity. Similar conclusion also made to the steel grade of the end-plate.

2.7.5 Non-linear Analysis of symmetric Flush end plate Bolted beam to column Steel Connection.[10]

The objective of the master project was to produce the moment rotation curve of the end plate connection using LUSAS software. The Finite element analysis result was compared with existing data that available from previous experiment. His model was based on that generated by Jim Butterworth. The element HX8M for the solid material, QTS4 and TTS3 for the surface element, BRS2 for the shank of the bolts and JNT4 for the contact element between the end plate and face of the column flange.

2.7.6 Pretasi sambungan pada paksi major menggunakan kerata keluli tempatan. [2]

He tested seven experiments on flush end plate and nine experiments on extended end plate. The column and beam sections were based on the local Perwaja section. All the properties of the sections was tested and included in his report. He discussed about the failure modes of the specimens and estimated the moment resistance of each connection based on the moment rotation curve.

CHAPTER III

RESEARCH METHODOLOGY

3.1 Introduction.

In this chapter we discussing the method used to develop the moment rotation curve is discussed. There are several way to obtained the M-ϕ curve. The method are curve fitting method, simplified analytical models, mechanical models, experiment in laboratory and finite element method. In this master project report discuss about the finite element method only which used the LUSAS Modeller.

3.2 Finite Element Model

A model geometry is entered in terms of features which are sub-divided (discretised) into finite elements in order to perform the analysis. Increasing the discretisation of the features will usually result in an increase in accuracy of the solution, but with a corresponding increase in solution time and disk space required. The features in LUSAS form a hierarchy that is Volumes are comprised of Surfaces, which in turn are made up of Lines or Combined Lines, which are defined by Points.

A model is created using command files rather than the CAD interface tools even though it takes longer time and initially tedious. The command file can simply be copied and edited. The command file is also well described by 6 comments within the file to provide a complete history of the model creation. Finite element models can often be a black box that provides answers without the user being fully aware of what the model exactly entails. The extra work in creating the command files has been well worth the effort and allowed the subsequent models to be created quickly. The technique of finite element lies in the development of a suitable mesh arrangement. The mesh discretisation must balance the need for a fine mesh to give an accurate stress distribution and reasonable analysis time. The optimal solution is to use a fine mesh in areas of high stress gradients and a coarser mesh in the remaining areas. To further reduce the size of the model file and the subsequent processing time symmetry was employed. The connection arrangement was symmetrical about a vertical centerline and therefore viewed from 0,0,0 only the right hand side was modeled.

3.3 Constitutive Models

3.3.1 Isotropic Model

For isotropic linear elastic materials, two materials properties are utilized in forming the element modulus matrix such as Young's modulus and Poisson's ratio. The material properties remain constant through out the analyses unless temperature dependent material assignment are utilized with varying temperature field.

In this research, thick shell element was found to be most suitable to idealize column web, beam web associated with isotropic properties. This was based on previous research which had shown that this could give good result. Further information about the element types used can be obtained in section 3.4.

3.3.2 Stress Potential Model

In LUSAS, the isotropic material non-linearity can be defined through several material model and each of the model have their own special characteristics. Stress potential model is one model under isotropic material which is used to describe the non-linearity of the material. For this application, non-linearity material properties applicable to general multi-axial stress state will be used and this will require the specification of yield stresses in each direction of the stress space when defining the yield surface. A number of special properties has to be defined in this stress potential model including yield stress, heat fraction (optional) and hardening properties.

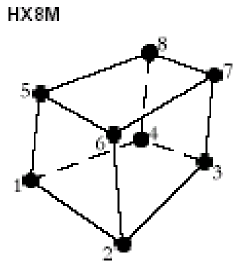
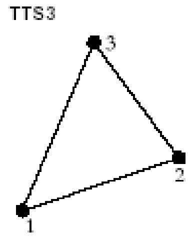
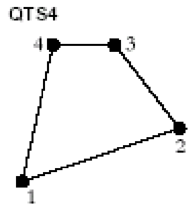
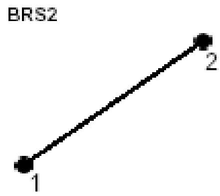
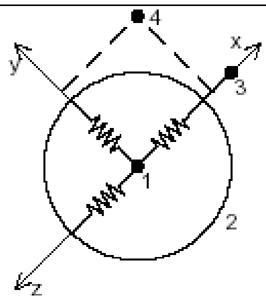
In material properties, initial uni-axial yield stress has to be defined as well to indicate the initial progress of plastic behaviour of such material. Heat fraction only applicable to temperature dependent materials and coupled analyses where heat is produced due to the rate of generation of plastic work. Heat fraction was not included in this study because temperature effect was not be considered. The other significant properties related to this stress potential model is hardening properties. Hardening properties was an important issue in this study as it was predicted to occur stage of non-linearity.

There are three methods for defining nonlinear hardening in LUSAS. Hardening curve can be defined in terms of either the hardening gradient, the plastic strain or the total strain. It is important to specify the slope of the stress strain curve up to yield stress in terms of the Young's Modulus. Plastic properties like the yield stress should be specified and the hardening data could be input as a series of coordinates.

3.4 Types of Element

The element used in this study are expected to provide acceptable good analytical results compared to the experimental results. Since the analysis carried out in this research consists of 3D analysis, therefore all elements used were in the group of 3D as it could produce more detail output due to the complex behaviour of the structure. Table 3.1 below shows the summary of the element types used in this research.

Table 3.1 The Element types used to the model

Section	Element type	Description of element	
Column Flange near to end plate, End plate, bolt/nut and beam flange	HX8M	three dimensional solid hexahedral elements comprising 8 nodes each with 3 degrees of freedom	
Column Flange, beam web, column web,	TTS3 and QTS4	three dimensional flat facet thick shell elements comprising either 3 or 4 nodes each with 5 degrees of freedom	 
Bolt/shank	BRS2	three dimensional bar elements comprising 2 nodes each with 3 degrees of freedom	
joint	JNT4	non-linear contact gap joint elements and are used to model the interface between the end plate and the column flange	

3.5 Load and boundary Condition

Base on the experiment reported by Husin[2], the boundary condition at the column top was put as roller where all nodes are not allowed to move along the x and z axes but rotation was allowed about the y direction. All node at the bottom of the column were assumed to be in restrained from moving in x, y and z direction, but rotation is allowed about x, y, z directions. Displacements in the x direction were restrained at all nodes on the plane of symmetry of the model. An initial point load was placed at 1.3 m from the face of the column flange.

3.6 Non-Linear Analysis

Non-linearities may arise in several forms including large deflections, large straining, non-linear stress-strain laws, deformation dependent boundary conditions and deformation dependent loading.

3.6.1 Materially Non-linear Analysis

This type of analysis is utilized if the material stress-strain relationship is significantly non-linear. Consider for example the idealized stress-strain relationship for a steel bar as shown in Fig. 2.21. This is linear in the elastic range so that an elastic analysis would predict the correct deformed configuration provided the yield stress is not exceeded. If yielding occurs, then the stiffness of the bar decrease resulting in a non-linear stress-strain law. Therefore incremental loading is required to trace the complete material response. LUSAS has a number different material models which permit modeling of the variety of physical materials including ductile metals, concrete, foam and soils.

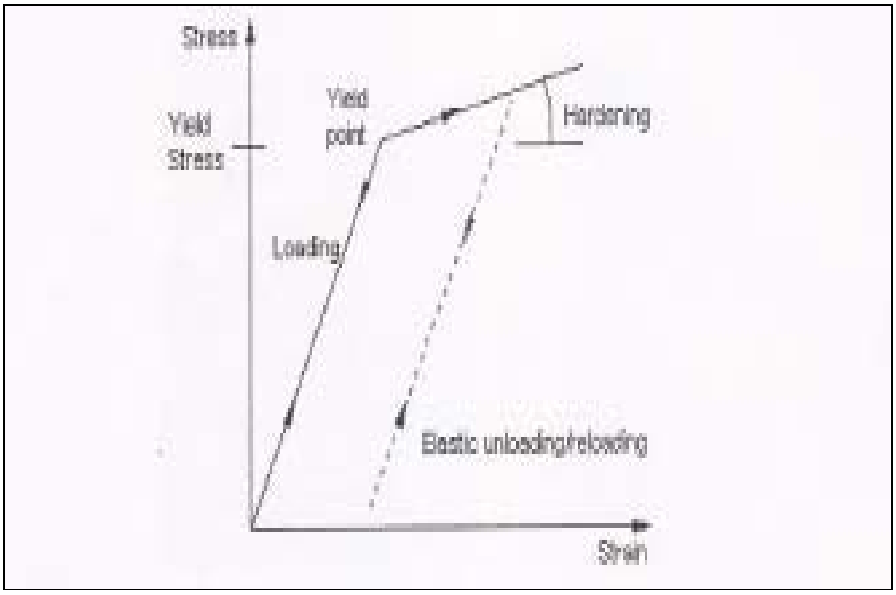


Figure 3.1:.Typical idealized uni-axial stress-stain relationship for steel.

3.6.2 Geometrically Non-linear Analysis

In this analysis, the changing effect of structural deformation on the structural stiffness and on the position of applied load is considered. A simple problem that illustrated this is a simply supported beam with uniformly distributed loading as shown in Fig. 3.2. The linear solution would predict the familiar simply supported bending moment and zero axial force. However, in reality, as the beam deforms so the angle of inclination of the beam at the supports introduces an axial component of force. This force may become significant if the deformations and consequently the angle of inclination become large.

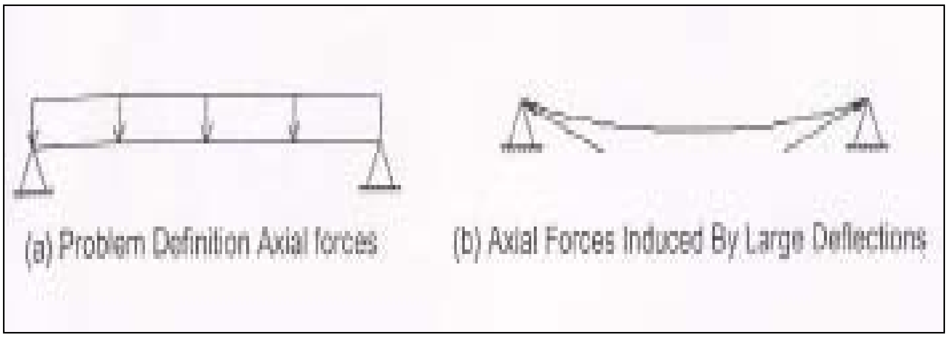


Figure 3.2 : Geometric Non-Linearities

3.6.3 Non-linear FEA Procedures

The non-linear procedure can be summaries into several steps such as:-

(a) Geometric modelling and discretized into finite elements (convergence study)
(b) Formulating finite element: geometric characteristic and data input-output.
(c) Material response characteristics for element(material properties, concrete, steel etc.)
(d) Properties assigned to finite element models (solid, planar or line, etc)
(e) Calibrating model and checking at linear state.
(f) Force increment loading or displacement increment modelling.
(g) Numerical modelling i.e Newton Raphson, arc length.
(h) Computational modelling i.e matrix solution scheme.
(i) Termination of non-linear finite element analysis.
(j) Non-linear finite element analysis out-put and result interpretation.

For nonlinear analysis, since it is no longer possible to directly obtain a stress distribution which equilibrates a given set of external loads, a solution procedure is usually adopted in which the total required load is applied in a number of increments. Within each increment a linear prediction of the non-linear response is made, and subsequent iterative corrections are performed in order to restore equilibrium by the elimination of the residual or 'out of balance' forces.

The iterative corrections are referred to some form of convergence criteria which indicates to what extent an equilibrate state has been achieved. Such a solution procedure is therefore commonly referred to as an incremental-iterative method shown in Fig.3.3. In LUSAS, the non-linear solution is based on the Newton-Raphson procedure. The details of the solution procedure are controlled using the non-linear control properties assigned to load case. For the analysis of non-linear problems, the solution procedure adopted may be of significance to the results obtained. In order to reduce this dependence, wherever possible, nonlinear control properties incorporate a series of generally applicable default settings and automatically activated facililities.

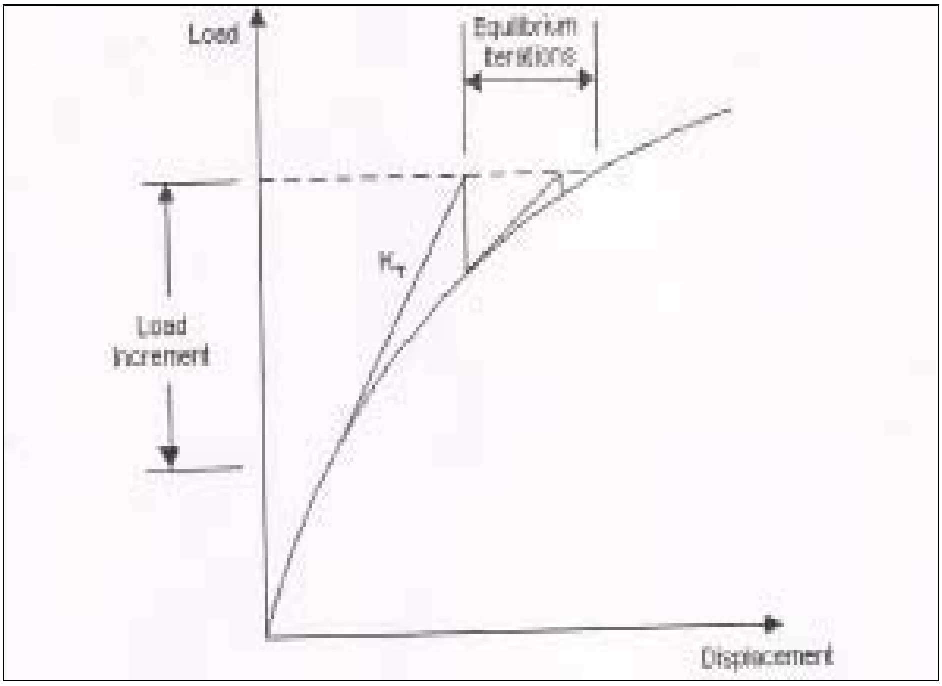


Figure 3.3: Increment iterative method

3.6.4 Increment Procedure.

For the Newton-Raphson solution procedures it is assumed that a displacement solution may be found for a given load increment and that, within each load increment, the load level remains constant. Such methods are therefore often referred to as constant load level incrementation procedures. However, where limit points in the structural response are encountered (for example in the geometrically nonlinear case of snap-through failure) constant load level methods will, at best, fail to identify the load shedding portion of the curve and, at worst, fail to converge at all past the limit point. The solution of limit point problems therefore leads to the development of alternative methods, including displacement incrementation and constrained solution methods.

3.6.5 Incremental Loading

The choice and level of incrementation will depend on the problem to be solved. Incrementation for non-linear problems may specified in four ways which are:

(a) Manual incrementation where the loading data in each load increment is specified separately.
(b) Automatic incrementation where a specified load case is factored using fixed or variable increments.
(c) Mixed incrementation mixed manual and automatic incrementation
(d) Load Curves where the variation of one or more sets of loading data is specified as load factor vs time load curve.

3.6.6 Solution termination

When using manual incrementation, the solution will automatically terminate following execution of the one increment. With automatic incrementation, the solution progresses one non-linear Control chapter at a time. The finish or each Non linear control chapter is controlled by its termination parameters. Termination may be specified in 3 ways, which are:

(a) Limiting the maximum applied load factor.
(b) Limiting the maximum number of applied increments
(c) Limiting the maximum value of a named freedom.

3.6.7. Non linear Convergence Criteria

The convergence criteria specify to what extent the numerical iterative procedure has reached the true equilibrate state. The specification of convergence therefore involves two considerations, which are type of convergence criterion and convergence tolerance. The types of convergence criteria incorporated in LUSAS are as follows:

(a) Absolute residual norm
(b) Root mean square residual norm.
(c) Displacement norm
(d) Residual force norm
(e) External work norm
(f) Incremental displacement norm.

The convergence tolerance for each criterion is specified in the solution parameters and advanced solution parameters section of the non-linear control properties. The selection of the convergence criteria and the associated tolerance is problem dependent. However, there are certain points should be considered. Clearly, the convergence criteria must not be too slack so as to yield an inaccurate solution, nor too tight so as to waste computer time performing unnecessary iterations. In general, sensitive geometrically non-linear problem require tight convergence criteria, where with predominantly materially non-linear problems, larger local residuals may be tolerated.

CHAPTER IV

RESULTS AND DISCUSSIONS

4.1. Introduction

The main aim of this research is to plot the Moment-Rotation(M- ϕ)curve for the flush end plate using finite element analysis. Non-linear analysis of flush end connection was successfully analysed using the LUSAS modeller software.

4.2 Non-linear Analysis Result.

4.2.1 Dimensions and Geometric Parameters

The actual model generated using LUSAS is shown in the Fig. 4.1.

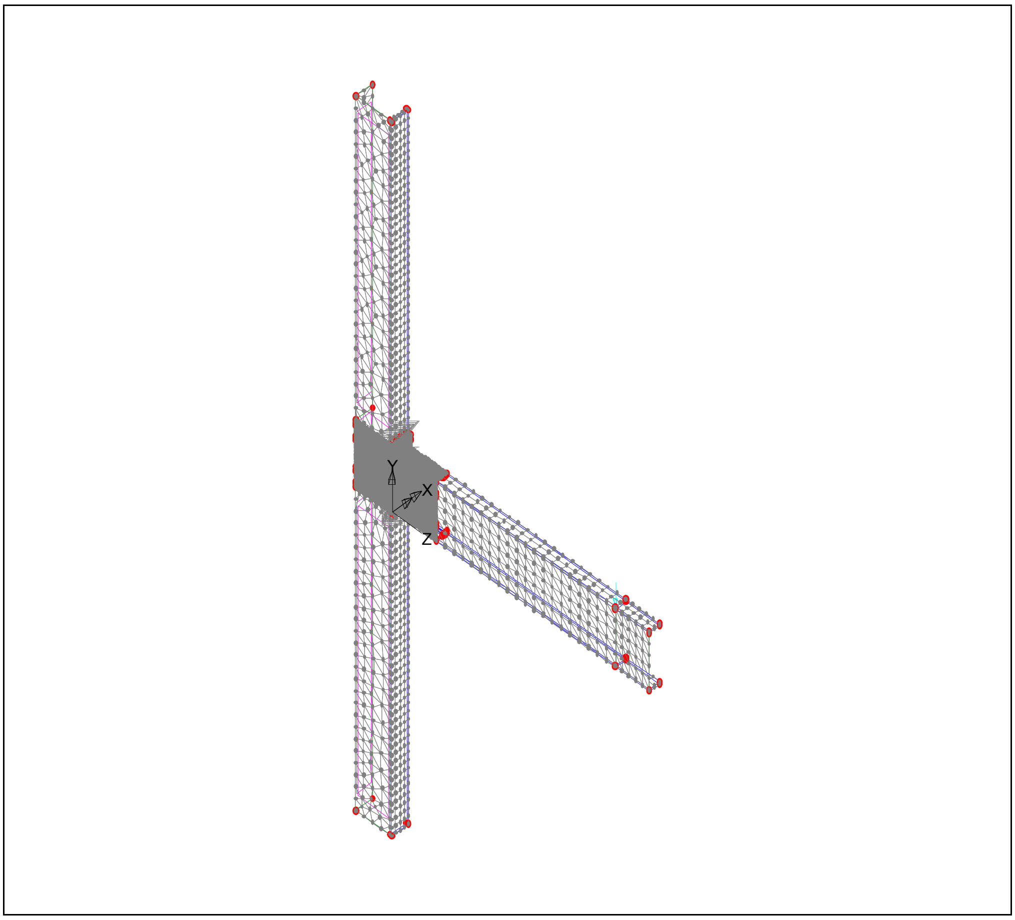


Figure 4.1: Model Geometric

The dimensions of this model are exactly the same as those of the flush end plate tested by Husin [2] are shown in Table 4.1 and poisson ratio was taken as 0.3 for all.

Table 4.1: Dimensions (in mm) of the flush end plate connection.

Element	Young Modulus(E) (kN/mm^2)	Yield stress fy (N/mm^2)	Ultimate stress,fu (N/mm^2)	*Thickness* *t* (mm)
Column (200 x 200 x 56.1)				
flange	194	367	528	12
web	198	385	547	12
Beam (250 x 125 x 25.1)				
flange	208	388	521	8
web	193	356	506	5
End plate	203	305	467	12

From the analysis, we obtained the deflected shape of the model due to the applied load of 1250 N. The results of interest in this analysis is the displacement of the beam and column in order to calculate the rotation, ϕ, of beam and column.

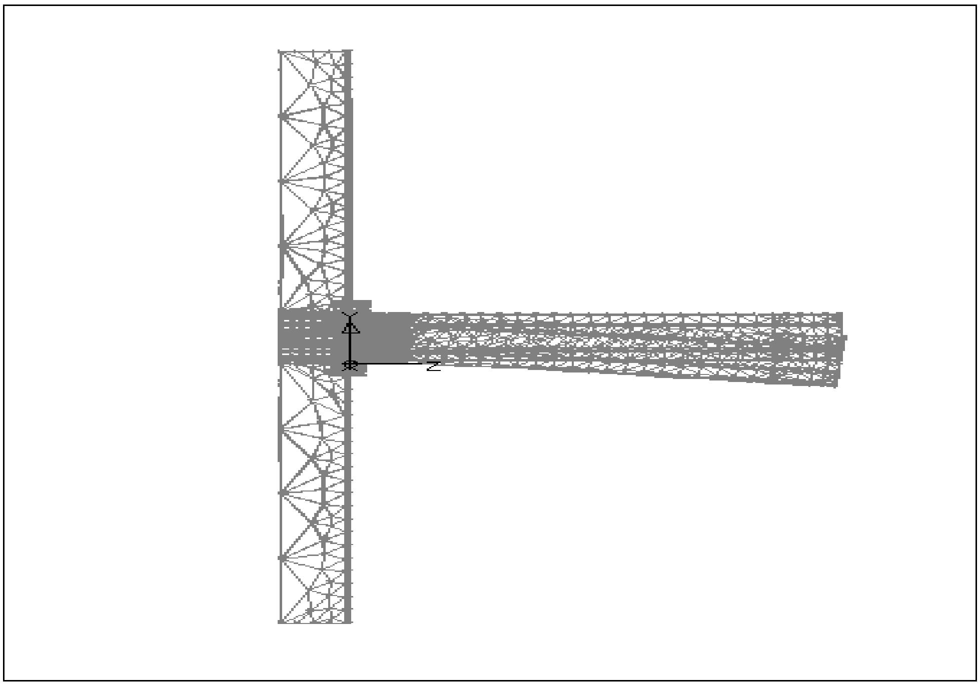


Figure 4.2 : Deflected shape of the overall model

Displacements at two location were recorded for each load increment (Fig. 4.2) beam displacement position at the inclinometer position (180 mm from face of column) in the laboratory and the second location is along the column web centre line which is 140 mm above the point of intersection of the beam web and column web centreline.

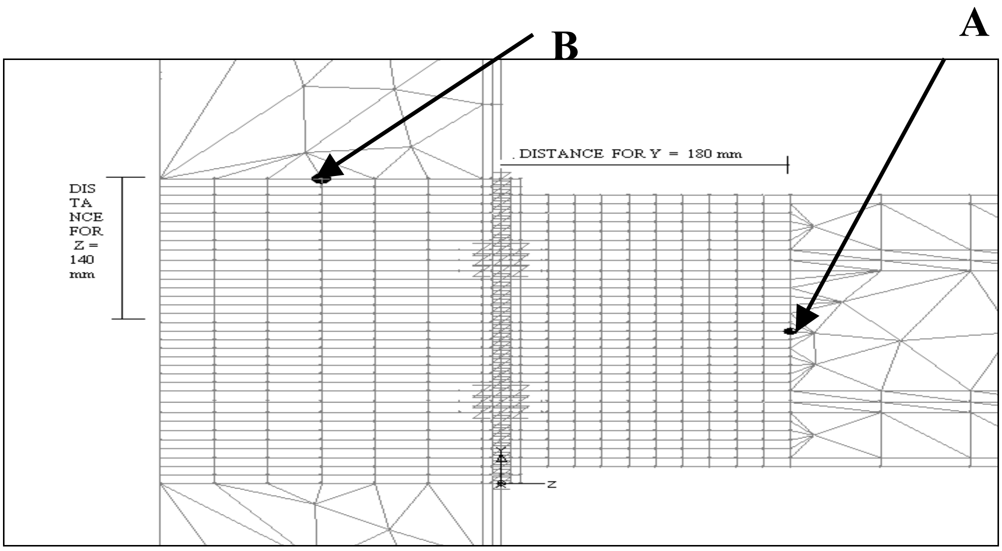


Figure 4.3 : Point A and B for Displacement.

The rotation of the connection can be calculate base on the displacement of beam (point A) in Y direction and displacement of the column (point B) in the z direction. The connection rotation, ϕ_j, is defined as

$$\phi_j = \phi_b - \phi_c$$

where: ϕ_b = beam rotation $(\tan^{-1} (\Delta y / 180)$

ϕ_c = column rotation $(\tan^{-1} (\Delta z / 140)$

Δy = Vertical displacement

Δz = Horizontal displacement

The applied moment can be calculated by the formula below

Moment = Total load factor x 1.25 x 1.3

Where M is applied moment in kNm and 1.25 is the initial point load in kN and 1.3 is the distance from the face of column flange in metre.

All displacement are tabulated in table 4.2.

Table 4.2 : Tabulated displacement for points A and B

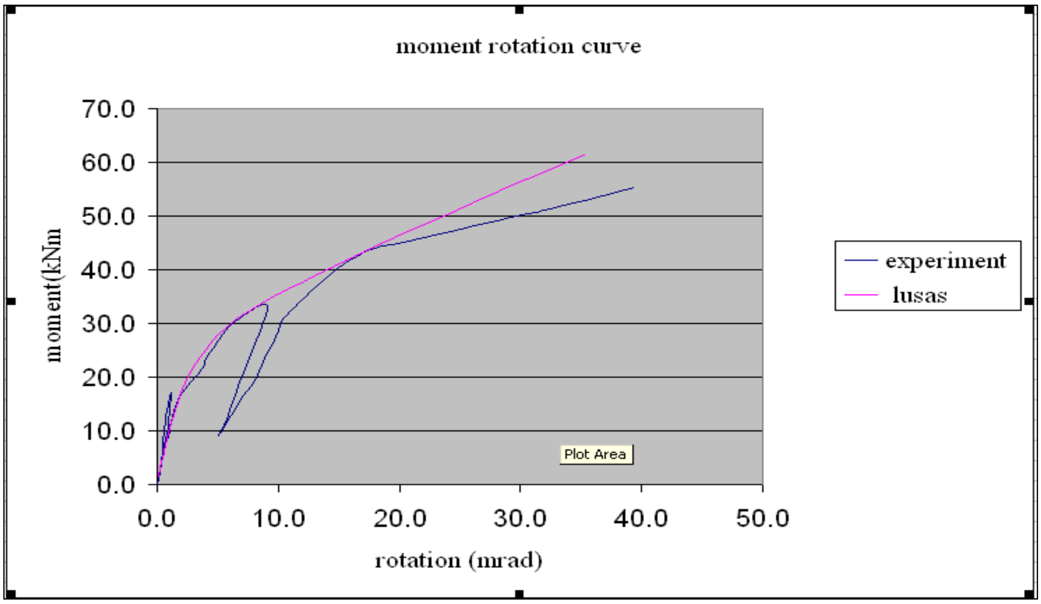
Load factor	Displacement(Δ y) (Of point A) /mm	Displacement(Δ z) (Of point B) /mm
1	-0.09559	0.049096
3	-0.28676	0.147288
6	-0.58275	0.295362
9	-0.90211	0.444604
12	-1.25972	0.595298
15	-1.72496	0.747771
18	-2.29388	0.904762
21	-3.16208	1.133044
22.89121	-3.94236	1.370304
25.02518	-4.96343	1.715828
27.15368	-6.07939	2.116397
29.39061	-7.3481	2.592019
32.27547	-9.08326	3.262549
34.99889	-10.7829	3.938645
37.93434	-12.6761	4.711811

Table 4.3: Tabulation of Moment and Rotation(ϕ_j)

Moment (kNm)	Δ y (mm)	Δ z (mm)	ϕ_b (mrad)	ϕ_c (mrad)	ϕ_j (mrad)
1.625	0.000531	0.000350687	0.531029	0.350687	0.180343
4.875	0.001593	0.00105206	1.593089	1.052061	0.541029
9.75	0.003238	0.00210973	3.237536	2.109733	1.127803
14.625	0.005012	0.003175742	5.011789	3.175753	1.836036
19.5	0.006998	0.004252125	6.998551	4.252151	2.7464
24.375	0.009583	0.005341218	9.583407	5.341269	4.242138
29.25	0.012744	0.006462586	12.74446	6.462676	6.281789
34.125	0.017567	0.008093172	17.56893	8.093349	9.475582
37.19822	0.021902	0.009787884	21.9055	9.788197	12.11731
40.66592	0.027575	0.012255913	27.58163	12.25653	15.3251
44.12472	0.033774	0.015117123	33.78727	15.11827	18.66899
47.75973	0.040823	0.018514424	40.84548	18.51654	22.32894
52.44763	0.050463	0.023303922	50.50546	23.30814	27.19732
56.8732	0.059905	0.028133176	59.97692	28.1406	31.83631
61.64329	0.070423	0.033655791	70.53944	33.66851	36.87093

Table 4.3 shows the calculated moment-rotation values. These results are then plotted in Fig. 4.4.

The LUSAS M-ϕ graph was then superimposed with the experiment result (from reference [2]) for comparison. Graph below shows the Moment rotation curve for experiment and the LUSAS output.



The graph is base on the spring stiffness of k = 1E9 N/mm for the end plate/column flange interface and k =1E12 N/mm for the bolt/end plate(bolt/column flange) which is the best curve that coin-side with the experiment. It can be seen that both graphs start initially as straight lines which indicate that the connection is initially is elastic. Then, with increasing loads, both graphs turn plastic. If comparison between these two graphs with that of connection 5 (in fig. 2.9), it can be concluded that the connection used in the present study is term as a semi rigid connection.

Using the "knee method method" , the value of moment of resistance M_R for the experiment and LUSAS curve are 37 KNm and 32 KNm respectively.

Therefore there is a difference of about 15.625 % between the two sets of M_R values.

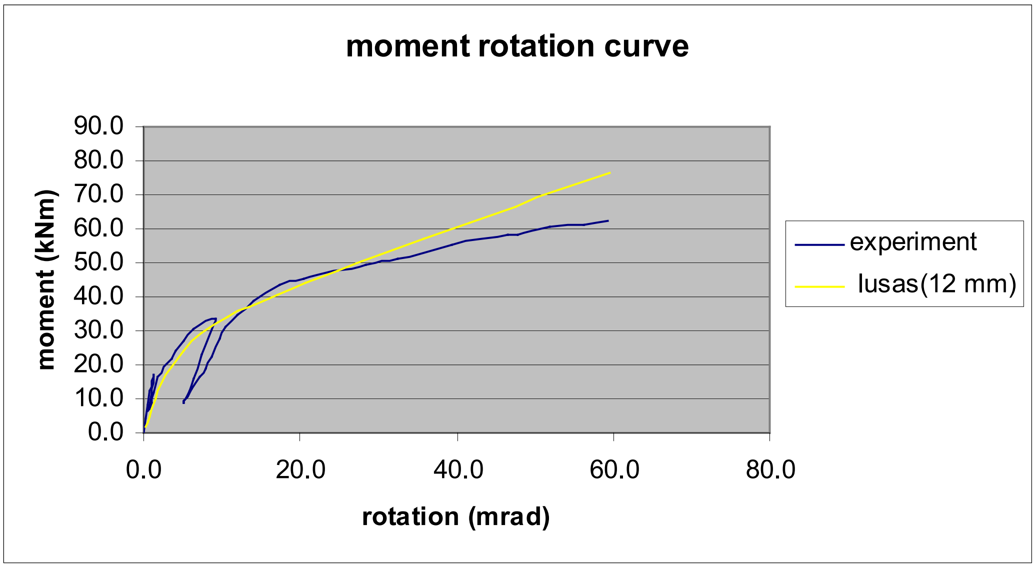


Figure 4.5: Graph M- ϕ_j

A second analysis was performed by increasing the value of the spring stiffness for the end plate/column flange interface as well as for the bolts/end plate interface.

The LUSAS M-ϕ graph in Fig 4.4 based on the spring stiffness of k = 10E6 N/mm for the end plate/column flange interface and k = 20E12 N/mm for the bolt/end plate(bolt/column flange). This is superimposed with the experimental M-ϕ curve.

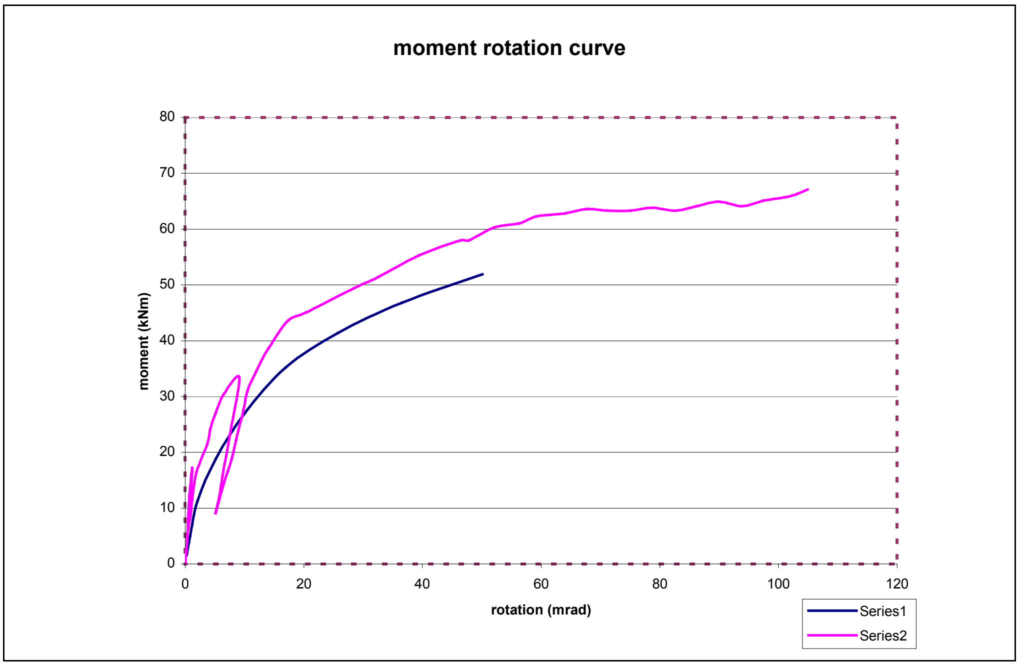


Figure 4.6 : Graph M- ϕ_j

A third analysis was performed by using relatively small value of spring stiffness for the endplate/column flange interface as well as for the bolt/end plate interface.

The LUSAS M-ϕ graph shown in Fig 4.6 is based on the spring stiffness of k = 1E3 N/mm for the end plate/column flange and k =10E3 N/mm for the bolt/end plate(bolt/column flange). This is then superimposed with experimental M-ϕ curve.

From the three graphs shown in fig. 4.3, 4.4 and 4.5, it can be seen that by decreasing the values of the spring stiffness of the end plate/column interface and the bolt/end plate interface, the LUSAS M-ϕ graph tend to diverse more and more from experimental M-ϕ graph. This provide and indication the spring stiffness of k =1E9 N/mm (for the end plate/column interface and k =1E12 N/mm (for the bolts/end plate interface) provide a good M-ϕ curve in comparison with that of the experimental M-ϕ curve.

Using these values of k, a fourth analysis was conducted to study the effect of increasing the end plate thickness from t= 12 mm to t=15 mm. The results of both cases are plotted as shown in Fig. 4.7.

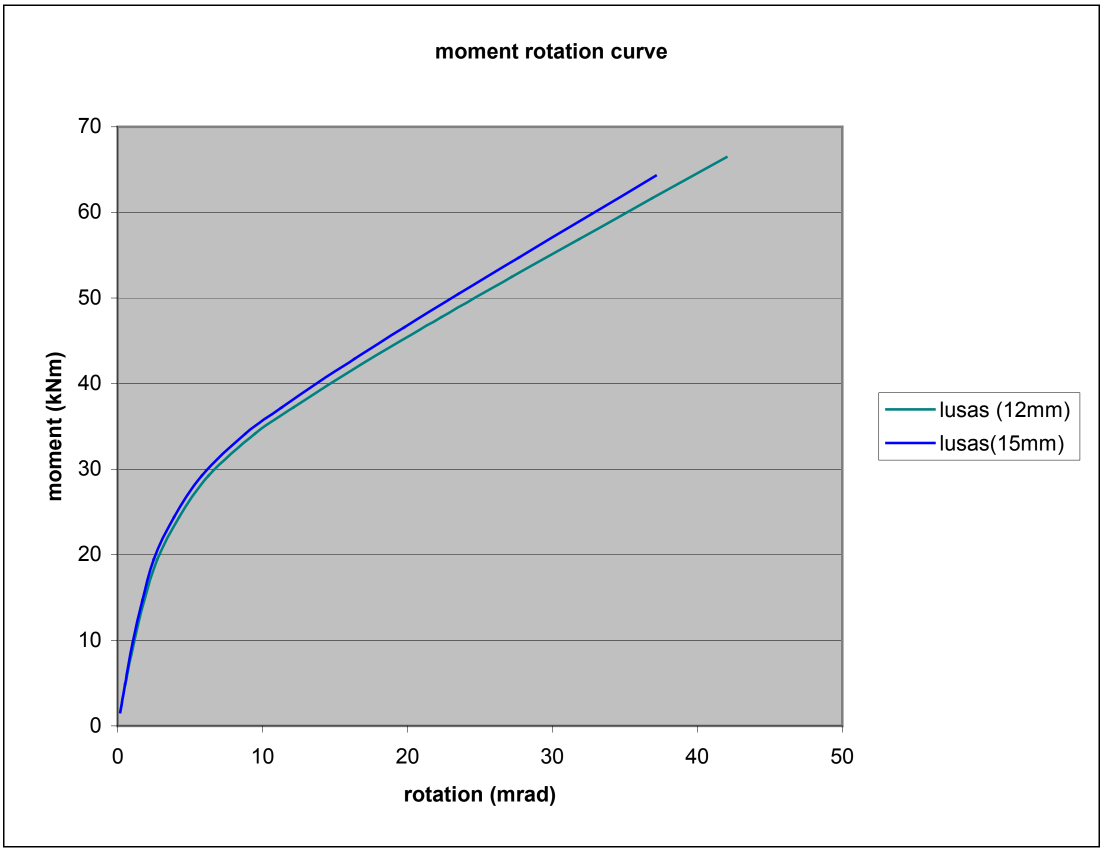


Figure 4.7 : Graph M - ϕ_j

From both graphs it can be seen that the stiffness of connection increase as the end plate thickness increasing. But the ductility of the connection is decreases slightly.

Using the “knee method” the values of moment of resistance, M_R for the 12 mm and 15 mm end plate thickness are 32 KNm and 33 KNm. Therefore it shows that a 3.125 % increase in the end plate produces a 3.03% increase in moment of resistance. M_R , of the connection.

CHAPTER V

CONCLUSION AND RECOMMENDATION FOR FUTURE RESEARCH

5.1 Conclusion

A finite element non linear analysis was carried out using LUSAS software to obtain the M-ϕ curve. From this study, it can be concluded that:

(a) A good M-ϕ curve of the flush end plate can obtained by using the LUSAS modeller with the appropriate value of spring stiffness i.e k = 1E6 N/mm for the end plate and column interface and k = 1E12 N/mm for the bolt to end plate/bolt to column flange interface.

(b) From the analysis the moment resistance, M_R for the flush end plate connection obtained by LUSAS was 32 kNm compared to the experimental value, which is 37 kNm. (using Knee Method)The difference is about 15.62%.

(c) Moment resistance of the connection change with changes in thickness of the end plate.
Connection with 12mm thickness end plate, M_R = 32.kNm.
Connection with 15mm thickness end plate, M_R = 33.kNm.

5.2 Recommendation for future reference.

There are several suggestion for future studies;

(a) Other softwares such as ABAQUS or COSMOS-M can be used to obtain the M-ϕ curve for other type connection .

(b) This research should be done in parallel with the experiment in the laboratory so that data required for computer input is available.

(c) Stress/strain curve should be recorded for material which required in interpret hardening slope for input in analysis.

(d) Convergence study should be carry out in order to obtain the optimum meshing in the modelling process.

LIST OF REFERENCES

1. FEA Ltd, "LUSAS: Modeller User Manual, Version 13", United Kingdom.

2. Md. Azman Bin Hj. Husin. "Prestasi Sambungan Pada Paksi Major Menggunakan Keratan Keluli tempatan'. UTM, January, 2001.

3. W. F Chen, Yoshiaki Goto, J.Y. Richard Liew, ' Stability design of semi rigid Frames., John Wiley & Sons Inc, 1996

4. Bose B, Sarkar S, and Bahrami M, Finite Element Analysis of unstiffened extended end plate connections, Structural Engineering Review, 3, 211-224, 1991

5. Bose B, Youngson G K, and Wang Z M, An appraisal of the design rules in Eurocode 3 for bolted end plate joints by comparison with experimental results, Proceedings from the Institute of Civil Engineers Structures and Buildings, 1996

6. Jim Butterworth, "Finite Element Analysis of Structural Steelwork Beam to Column Bolted Connections', Constructional Research Unit, School of Science & Technology, University of Teeside, UK

7. Y.I.Maggi, R.M.Goncalves, R.T.Leon, L.F.L.Ribeiro, 'Parametric analysis of steel bolted end plate connections using finite element modeling', Journal of Constructional Steel Research 61, Elsevier, 2005, pg 689-708

8. Ali Abolmaadi, John H Malthys, Mohammed Frooqi, Yeoh Choi, 'Development of moment-rotation model Equations for flush end-plate connection', Journal of Constructional Steel Research, Elsevier, 2005.

9. Ana M. Girao Coelho, Frans S.K Bijlaard, luis Simoes da Silva, ‘ Experimental assessment of the ductility of extended end plate connection’, Journal of Constructional Steel Research 26, Elsevier, 2004, pg 1185 –1206.

10. Che Husni Ahmad. “Non-linear Analysis of symmetric Flush End Plate Bolted Beam to Column Steel Connection”. UTM, November, 2005.

11. Emmett A. Summer, Timothy W. Mays, Thomas M. Murray ‘End plate moment connections: Test results and finite element method validation’, Department of Civil E and Environmental Engineering, Virginia Polytechnic Institute Blacksburg, Virginia, USA.

12 .A.Smanson, D.S.Kokan, R.T.Leon, ‘Advanced finite element modeling of bolted T-stub connection components’, Journal of Constructional Steel Research 58, Elsevier, 2002, pg 1015 – 1031